# NUMERICAL INVESTIGATIONS OF FLOODING FOR TWO-PHASE COUNTER-CURRENT FLOW

*A Thesis submitted in partial fulfillment of the requirements for the Degree of*

**Bachelor of Technology**

in

**Mechanical Engineering**

by

**Atul Dewangan (Roll No. 110ME0291)**

under the guidance of

**Dr. Suman Ghosh**

**Department of Mechanical Engineering**
**National Institute of Technology**
**Rourkela – 769008**

NATIONAL INSTITUTE OF TECHNOLOGY ROURKELA

# CERTIFICATE

This is to certify that the research work that has been presented in this thesis entitled "**NUMERICAL INVESTIGATIONS OF FLOODING FOR TWO-PHASE COUNTER-CURRENT FLOW**" by Atul Dewangan (Roll No.110ME0291), has been carried out under my supervision in partial fulfillment of the requirements for the degree of Bachelor of Technology in Mechanical Engineering during session 2013-2014 in the Department of Mechanical Engineering, National Institute of Technology, Rourkela.

To the best of my knowledge, this dissertation work has not been submitted in any other college or university at any time prior to this, for the award of any degree or diploma.

Place: Rourkela
Date:

**Dr. Suman Ghosh**
Assistant Professor
Department of Mechanical Engineering
National Institute of Technology, Rourkela

# ACKNOWLEDGEMENT

I would like to thank my guide Dr. Suman Ghosh for providing me a golden opportunity to work under his esteemed guidance. It is only due to his guidance that this work has seen this day.

A special mention of Mr. Kumar Samal is a must for helping me even in minute matters in the project work playing the dual role of a friend and a guide at the same time.

Also, my parents always provided me with the best they could which imbibed a moral support in me due to which I and in turn, this work could withstand the test of time.

Atul Dewangan

110ME0291

# CONTENTS

# INDEX OF FIGUREURES

# ABSTRACT

For certain values of fluid velocity of two phases, a wave forms covering as well as blocking the area of cross-section of the closed conduit of two phase flow. This phenomenon is called **Flooding**. The best example is the case of leakage in the essential cooling circuit of the reactor. This is highly undesirable and risky as it may be detrimental in the functioning of the system (for example- a nuclear explosion). Hence, efforts are being made to correctly predict the occurrence of flooding phenomena in nuclear reactor plant. It has been established from the work done in flooding that there is a lack of a universal parameter for its determination. The aim of the current allotted task is to numerically investigate phenomena of flooding in the duct of the hot power leg in nuclear reactor. The situation of the on-set of flooding is predicted and analyzed for different initial depth of water in duct of hot power leg using VOF model with Finite volume method for its prediction.

**Keywords:** Flooding, Counter-current flow limitation, Multiphase flow, Finite Volume Method (FVM)

# CHAPTER 1: INTRODUCTION AND LITERATURE REVIEW

This section begins with a brief introduction of the title followed by a literature survey which has been dealt in a very detail manner. Further, the gaps in the literature survey have been pointed out .The last section deals with the aims and objectives of the current work.

## 1.1 INTRODUCTION

Multiphase flow finds applications in instruments of everyday use. Even in the simplest of systems such as a closed conduit flow, single phase flow is rarely observed. Thus, it is very important to study each and every aspect of multiphase flow. In this regard, the focus is being laid on phenomenon of flooding which is generally observed in primary circuits of nuclear reactors. In a two-phase counter-current flow, flooding can be expressed as the blocking of one phase (may be lighter or denser) by the another, reason being the latter phase occupying the whole area of cross-section of the closed conduit due to formation of a wave.

## 1.2 PHENOMENA OF FLOODING

The flow of a gas and a liquid in counter-current sense is experienced in large portion of mechanical instruments, like, cooling systems drastically affecting their performance. The start of flooding inhibits the functioning of the above mentioned devices. The start of flooding is a central variable in the operational utilization of different sorts of instruments utilized as a part of industry and the expectation of flooding will help to shield numerous imperative modern requisitions. For certain values of fluid velocity of two phases, a wave forms occupying as well as choking the area of cross-section of the closed-conduit. Flooding plays a key role in the design of Pressurized-Water reactors. Pressurized Water reactors are so named due to the utilization of pressurized water in the essential cooling circuit of the atomic reactor. PWRs if encounter leakage in the essential cooling circuit are a source of a disaster for the surroundings. Thus, in case of LOCA, PWRs pose a serious threat of explosion to the population nearby.

## 1.3 LOSS OF COOLANT ACCIDENT

It is a term associated with PWRs in which the safety of primary circuit of the reactor is compromised. This leads to the formation of water-vapor in the primary circuit followed by the condensation of the vapor in the heat transfer bay involving both primary and secondary circuits.

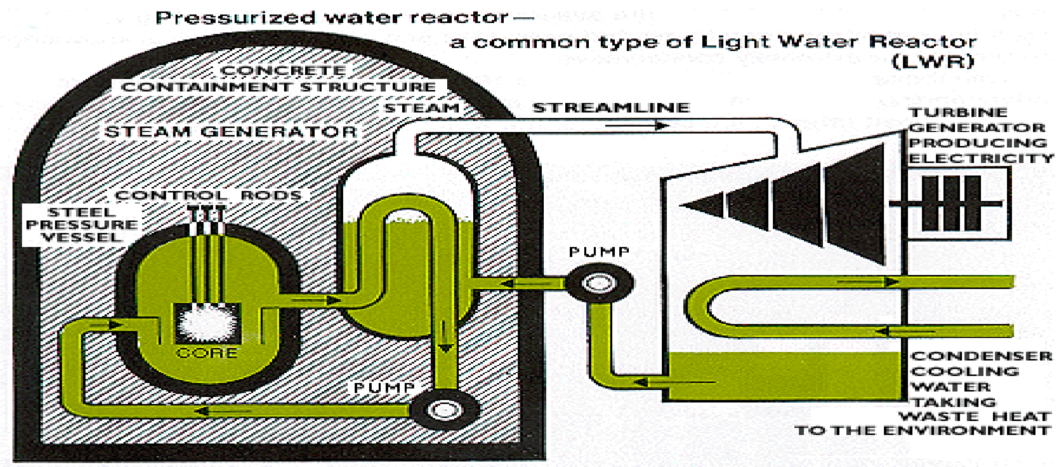



Figure 1.1: Schematic of a nuclear plant. (www.tbc.school.nz)

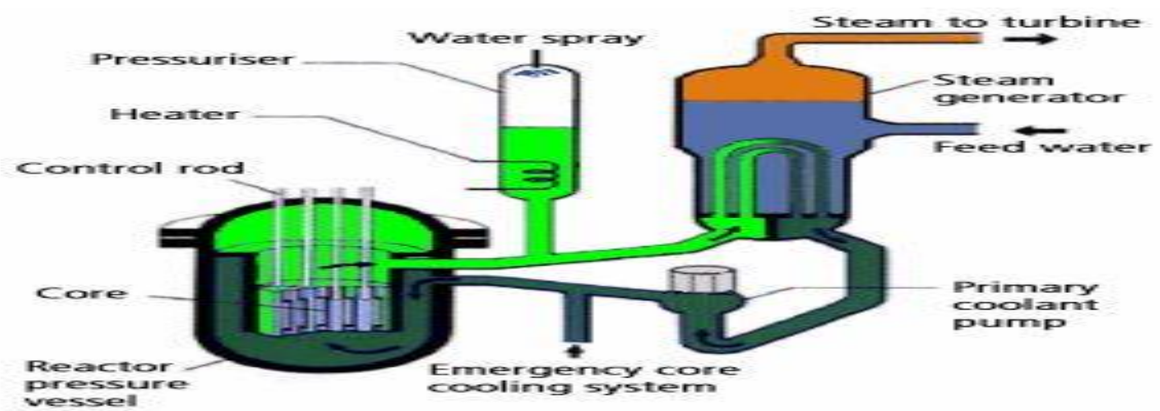



Figure 1.2: Nuclear power plant highlighting primary and secondary Cooling Circuits

In Figure 1.2, green (both dark and light) color represents the primary circuit and the blue color represents the secondary circuit. In case of LOCA, the pressurized water in the primary circuit is converted into vapor and as soon as it encounters the feed water of the secondary circuit, vapor condenses into liquid thus further blocking flow of vapor from the primary circuit to the heat transfer bay. This blockage can lead to an explosion which poses a serious threat to the nuclear reactor as well the population and the ecosystem surrounding it.

## 1.4. LITERATURE REVIEW AND GAPS IN LITERATURE

In this section, the literature review required for the present work has been dealt in a thorough manner as well as describing the shortcomings of the work done on the topic till date.

### *1.4.1 LITERATURE REVIEW*

The foundation for the almost all the current works in the field of multiphase flow could be followed to the vital work by **Wallis** *et al***.,** (1969). His work has paved a thousand ways in the field of multiphase flow. A ton of work has been carried out in the field of flooding and flow regime investigation. The measure of work in the area of counter-current multiphase flow is less as contrasted with that in co-current multiphase flow. **Lee and Bankoff** *et al.*, (1984) did a series of trial for the detection of flooding in a channel of rectangular shape utilizing a flow of water and steam in the counter-current sense. The conclusion drawn by the analyses made for detecting flooding was the ruling impact of the slight gap introduced in the channel on it. **Biage and Delhaye** *et al***.,** (1989) examined the instance of stream of air and water in a straight-up duct having a gap comparable to the dimensions of the channel. It demonstrated that the start of flooding is connected with entrainment of droplets. **Zapke and Kröger** *et al.*, (2000) contemplated the dependence of flooding phenomenon on the physical properties of the both gas and liquid and on geometry of the duct. The fluids used were water, propanol, methanol, argon, air, helium and hydrogen. It was inferred that the velocity of the gas during flooding emphatically relies on the tallness of the pipe and thickness of the liquid. **Deendarlianto** *et al.*, (2005) identified lower flooding and upper flooding. For lower flooding, the formation of wave occurs in lower part and it moves upwards. Lower flooding is supported by low value of rate of the flow of mass of liquid and greater inclination of the conduit. Similarly for upper flooding, the formation of wave occurs in lower part and it moves downwards enhanced by greater values of flow-rate of liquid and lower inclination of the conduit. The maximum value of wave occurs at an inclination of 45 degrees of pipe. **Drosos** *et al.*, (2006) did his trials to detect flooding for flow of gas and liquid in the counter-current sense on a straight-up restricted duct utilizing different fluids and air. For very small Reynolds number say (Re<250), the beginning or start of flooding is connected with flow-reversal, and disintegration of waves. **Ousaka** *et al.,* (2006) studied the dependence of the start of flooding on length of channel, diameter of channel, channel slant and the surface tension of the working fluid. He observed an inverse dependence of flooding velocity on the pipe length; flooding velocity takes a greatest quality at 45 degrees, diminishes with diminishing surface pressure because of the precise solid cooperation created between gas and fluid stages as an aftereffect of uncovered surface regions. **Deendarlianto** *et al.,* (2008) considered a hot leg in a PWR. Flooding was observed by measuring the height of water in the particular separators. The start of flooding happens simply with the framing of fluid slugs which create close to the twist of the hot force leg. The launch of flooding is profoundly influenced by the pressure of the system. More the value of pressure of the system, higher is the value net rate of air-flow required to start flooding. **Pantzali** *et al.,* (2008) likewise contemplated phenomena of flooding in little-breadth pipes (<10 mm). The nature or sort of flooding phenomena that is included with the slanted little measurement channels is wave transport in the upward direction by the gas. He additionally reasoned that rather than aggregate stream inversions as in the general case just a piece of fluid streams downwards. The utilization of surfactants prompts respectably defer of the start of flooding because of the substantial damping of wave amplitudes. **Trifonov** *et al.,* (2010) assessed the dependency of flooding on Reynolds number, distance between the plates and the physical properties of gases and liquids. The analysis was focused around the numerical solution of Navier-Stokes equation. **Deendarlianto** *et al.,* (2010) contemplated the impact of shifting surface strain in a slanted tube in two phase stream in counter current sense. For low surface strain, the predominant variable is droplet entrainment in the transport of fluid in the upward direction. The flooding sensation is almost autonomous of surface tension for low rates of fluid-flow. For flow at greater flow-rates of fluid, upper flooding converts to lower flooding because of the steady diminishing pressure at the surface. **Nishimoto** *et al.,* (2011) grasped the influence of the material of tube for studying flooding in a tiny diameter tubes (inner diameter <10mm). The critical flooding (beginning of flooding) and flow reversal are purely independent of the tube material. However, the tube material affects the flow patterns involved with flooding and the flooding velocity as calculated by the **Wallis** formula. **Trifonov** *et al.,* (2011) discovered two different workings behind gas/liquid flow in counter-current sense between creased plates. The former is concerned with lesser amplitude of corrugation which has a backing from gas-liquid flow .The second one is for the substantial qualities of layering sufficiency delineating where the free surfaces and the divider about touch one another. **Prayitno** *et al.*, (2012) concluded by the inspections on flooding in an almost flat pipe that flooding gas velocity has inverse dependence on the liquid flow rate, flooding velocity declines with the decrease in slant nature of the pipe, the consequences of using liquids of different viscosity is marginally more prevailing than the surface tension effect. **Miwa** *et al.*, (2013) considered the two-phase simultaneous flow of a gas and a liquid in counter-current sense in which the direction of flow of gas upwards and that for the liquid droplet flowing downwards was investigated. The droplets that account for the mass transfer are in the state of transition (flow-regime) and any chemical interaction between surrounding liquid phase and gas phase because of its exposure providing an enormous amount of surface area and thus, probability of a chemical reaction. The way of the droplet movement was analyzed to get to all obliged qualities for the fruitful outline of the two-phase flow framework in counter-current sense.

*1.4.2 GAPS IN THE LITERATURE REVIEW*

- Disregarding the expansive number of effects reported with respect to the flooding phenomena, there is still an extensive vulnerability in regards to the phenomenon at the start of flooding. Also, extra work must be carried out on stream in slender entries to support the information pool existing today and chalk out some precise tools for indicating flooding.

- It has been found through the studies on flooding that a planar surface of contact at the area of cross-section a tube, anticipates to a greater extent the normal free streaming fluid layer width at the base of the tube however it neglects to ascertain the width of the layer for the fluids of low viscosity within the study (e.g. butanol solution and tap water), where the state of interface is strictly concave.

- It is further emphasized that more information is obliged to cross-check the legitimacy of proposed connections proposed for effective forecast of the extent for the safe operational utilization of the instruments.

- As a rule, the issue is that the interrelations are not fit to anticipate the phenomena of flooding subject to the conditions that are completely not quite the same as the ones used to develop the correlations in any case.

- Additionally, the greater part of the papers gave to flooding are simply test estimations and perceptions. The hypothetical works with simultaneous flow of two phase in the counter-current sense with a wavy surface of contact is acknowledged are small in amount.

## 1.5 AIMS AND OBJECTIVES

The objective of the present work is to numerically study the gas-liquid flooding in hot leg of nuclear reactor using Finite Volume Method with Volume of Fluid model. The aim is to investigate the effect of height of water in the duct of hot power leg on flooding. It would serve as a reference guide in applications involving the use of flooding phenomena as a precautionary measure, one of the most important applications being the design of PWRs.

## 1.6 ORGANISATION OF THESIS

The entire thesis is divided into seven chapters:

**First chapter** introduces all the terms associated with our present work. The concepts and ideas that are required in the later part are being described in brief. It includes an in-depth literature survey undertaken the research done in the field of the flooding phenomena, gaps in the literature which gives an insight about the present scope. It provides the objective and a description to the overall work.

**Second Chapter** describes in-detail problem that is the center of attention in the present work.

**Third Chapter** comprises of the methodology adopted to approach the problem. It consists of Computational Fluid Dynamics; VOF Method and FVM both described in brief, Steps Followed in ANSYS**,** ***k-ω*** model turbulence model, Contour plots

**Fourth Chapter** is the description of results mostly in contour plot form and sometimes in the graph plots.

**Fifth Chapter** mostly deals with drawing conclusions based on the present work.

**Sixth Chapter** deals with the future recommendations based on the results obtained from the present work

**Seventh Chapter** gives a list of all the references used in the thesis.

# CHAPTER-2: PROBLEM DESCRIPTION

## 2.1 REPRESENTATION OF THE SET-UP

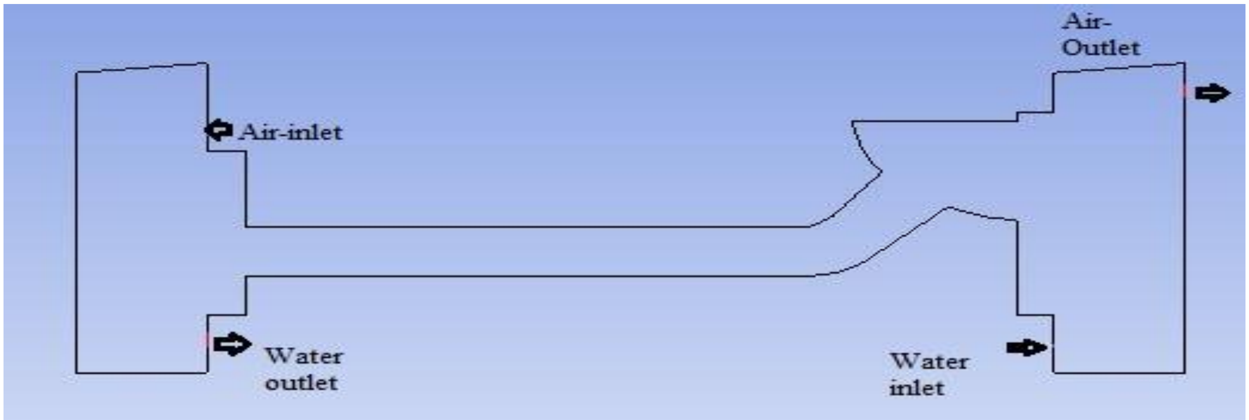
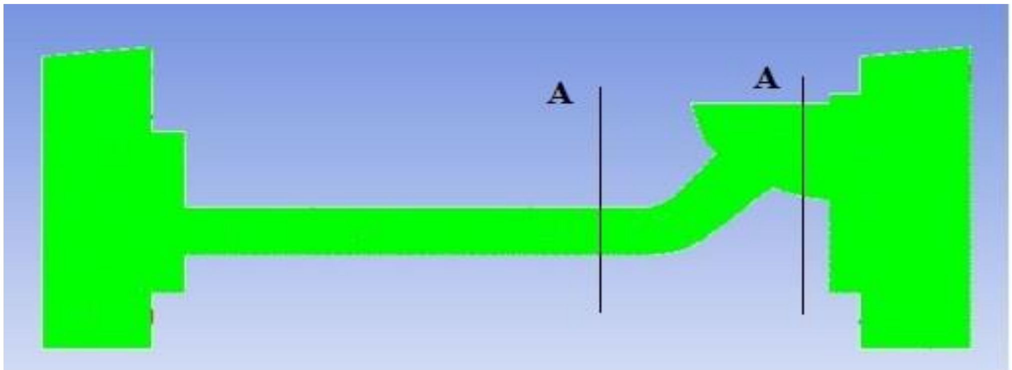



Figure 2.1: Schematic representation of the Problem in Consideration

The geometry of the present problem has been taken from **Deendarlianto** *et al.*, (2008). The launch of flooding is profoundly influenced by the pressure of the system. However, the present work deals with the consideration of amount of water present in the straight duct of the hot power leg measured by its tallness. The influence of the parameter considered on the start of flooding is the center of attention in our present study. This is a schematic outline representing hot power leg. The fundamental segments include a setup for the experiment to be done, including the pressure vessel in which the reaction takes place (RPV) simulator located on the left side

.

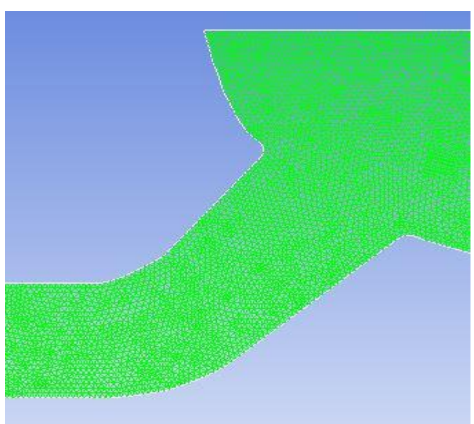
## 2.2 MESHING




Figureure 2.2: Meshed diagram of the hot power leg in consideration

**A-A Cross section**

The mesh element is triangular in nature and the size of the mesh (face mesh) is 0.01m. The meshing done for the problem has been displayed in Figureure 2.2 and Figureure 2.3.



Figureure 2.3: Grid pattern adopted for the investigation

## 2.3. BOUNDARY CONDITIONS

The boundary conditions are four in number.

(a)Air inlet (rate of flow of air = 0.183-0.274 kg /s)

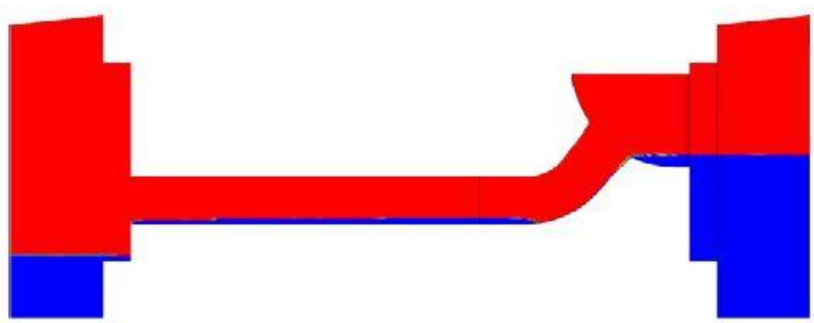
(b)Air outlet (constant pressure outlet, gauge pressure =0)

(c)Water inlet (flow-rate of air = 0.3 kg/s)

(d)Water outlet (constant pressure outlet, gauge pressure =0)

## 2.4. INITIAL CONDITIONS

The system pressure is taken to be 0.15 MPa. With the help of a software named DATAGRAPH, the height of the denser phase in the test setup was measured. It was later on patched in ANSYS to the mesh file before running the calculations. The initial water level in the test equipment is from the following image. This is a phase contour plot representing water and air. The red color in the image represents air and the blue color represents water.



Figureure 2.4: It represents a plot of the initial conditions in the hot power leg. Red color represents air while blue color represents water)

# CHAPTER-3: METHODOLOGY ADOPTED

The present work is purely based on study of two-phase flow through numerical analysis using ANSYS. The geometry of the hot power leg was created and meshed as shown in Figure 2.1 and Figure 2.2. The meshing of the geometry is done so as to minimize the error involved with the calculation of the results. The boundary and conditions used for the simulation run has been taken as described in section 2.3 and 2.4. The computational methodology that is being used to carry out the present work is a two dimensional analysis, transient analysis of the hot power leg as described in section 2.1. The numerical approach that is taken is the Finite volume method that is based on the VOF model both of which are described in details in sections 3.3 and 3.2 respectively. The files are recorded at regular intervals, every five hundred iterations.

## 3.1 NUMERICAL ANALYSIS

Computational fluid dynamics (CFD) deals with complex problems of fluid flows that cannot be solve analytically. The reason for the complexity may be the size of the control volume considered, impractical velocity requirements, excessive cost of a material etc. This is done with the help of numerical methods minimizing the error in the process. Some pre-defined algorithms such as SIMPLE, PISO are followed to get the optimized results for different cases. SIMPLE provides near to exact results when the problem is independent of time whereas PISO gives better results for transient calculations. The calculations are usually done on servers hosting a number of computers involving parallel computing sometimes even a supercomputer. The discretization methods that are generally used are Finite volume method, Finite differences method and Finite Element method. After discretization is done, focus is on the model used to extract the most appropriate result. Direct numerical simulations, large eddy simulations etc. are some of the examples.

CFD is an interdisciplinary science comprising Numerical analysis (mathematics), Fluid mechanics and Computer science. The advent of commercial CFD codes (CHAM, PHOENICS, ANSYS and many more) has totally revolutionized the design industry. The current generation of CFD packages generally is capable of producing accurate solution to simple flows and rarely used in very advanced research facilities due to accuracy limitations.

## 3.2 VOLUME OF FLUID (VOF) METHOD

It is a numerical method of solution for locating and tracking the fluid-fluid interface. It belongs to the category of Eulerian methods that are described by a stationary or a moving mesh to constantly predict the evolving shape of the interface. In other words, VOF is a scheme of advection, a numerical technique that enables the programmer to predict and track the position and shape of the interface, but it is not an independent flow solving algorithm. The Navier-Stokes equations representing the momentum transfer in the flow have to be solved separately.

## 3.3 FINITE VOLUME METHOD

It is a technique for representation and evaluation of partial differential equation PDE by changing them into algebraic equations. The finite volume method is based on Integration rather than Differentiation. "Finite volume" means the small volumes surrounding every node on a mesh. In the finite volume method, the volume integrals in a PDE that contain a term of divergence are converted to surface integrals, utilizing the Divergence theorem. Then, these terms are treated as fluxes at the cross-section of each finite volume. Since the flux that enters a given volume is equal to the flux leaving the adjacent volume, the method is conservative in nature.

## 3.4 STEPS FOLLOWED IN ANSYS

Step 1: File→Read→Mesh to read a mesh file

The first step is to read a mesh file in ANSYS.

Step 2: Mesh→Check

The meshing done previously is properly checked to minimize the error in the calculations. Mesh→Info has a lot of options to further validate the mesh.

Step 3: Define→General

The solver type is selected as pressure based, transient in nature, Two dimensional planar space and absolute velocity formation. Gravity is taken in the negative y direction as $9.81 m/s^2$.

Step 4: Define→Models.

The multiphase, viscous and energy models are switched on .The rest of them are left as it is. In the multiphase model, VOF model is selected, explicit scheme and the number of phases is two. Since the flow is turbulent *k-ω* model of turbulence is also selected.

Step 5: Define →Materials.

The materials used in the present work are air and water. These are already present in the ANSYS database and are selected by accessing the database from the pop-up window.

Step 6: Define→Phase

Air is selected as the primary phase and water as the secondary phase. The value for surface tension is fed in the system considering surface interactions between air and water.

Step 7: Define →Boundary Conditions

There are four boundary conditions. For air inlet, the value for air inlet is set to 0.268 kg/s and that for water is 0 kg/s. Similarly, for the water inlet the dimensions of the duct remain the same whereas the water flow-rate is set to 0.30 kg/s and that for air is 0 kg/s. The dimension of the both inlet ducts is 0.04 $m^2$ area of cross-section. The dimension of the outlet duct is 0.16m2. The boundary condition for both the outlets is pressure outlet.

Step 8: Solve→ Solution Methods

The solution method adopted is the PISO scheme of numerical evaluation. This is particularly helpful for transient calculations.

Step 9: Solve→Solution controls

The proper limit for every term is selected.

Step 10: Solve→ Monitors

Monitors play a very important role as they are instrumental in minimizing the error in the numerical analysis. The concerned factor here is the residual plot. Residual plots control the solution. The limit for both velocity and momentum is selected to be $10^{-6}$ and that of energy is $10^{-9}$.

Step 11: Solution→ Solution initialization

The solution is initialized from all zones

Step 12: Solution→ Run calculation to start the calculations

The time stop is takes as $10^{-4}$ seconds and the number of iteration is selected to be 20,000. The nature of time step is fixed

USING PATCH, REGION ADAPTATION

The region is adapted and combined at last to obtain the initial condition (i.e. Figure 2.4) Patch option enables a user impart the initial conditions to the geometry.

### 3.5 *k-ω* model

The *k-ω* model is a standout amongst the most usually utilized models of turbulence as a part of liquid stream. It is a model defined by two mathematical statements which implies that it incorporates two more transport comparisons to stay informed concerning the turbulent properties of the stream. It enables a two mathematical statement model to be answerable for past impacts like dissemination and convection of vitality in turbulent stream.

The initially transported variable is turbulent active vitality, 'k'. The second one is the particular dispersal, 'ω'. The latter decides the measure of the turbulence, while the first variable, 'k, represents the vitality in turbulence

# CHAPTER-4: RESULTS AND DISCUSSION

## 4.1 VALIDATION OF THE PREVIOUS WORK

In the initial part of the project work, a validation to study occurrence of the phenomena of flooding a nuclear reactor focusing on its hot power leg was done keeping mass rate of flow of water fixed at 0.3 kg/s and gradually changing the flow-rate of air from 0.183-0.274 kg /s. It was discovered that flooding phenomena relies on upon the rate of flow of the air. For higher flow-rates of air, flooding was watched soon enough as contrasted with low flow rates.This was well in accordance with the experimental results that we have beforehand. Below here are some of the plots to prove the above statement.

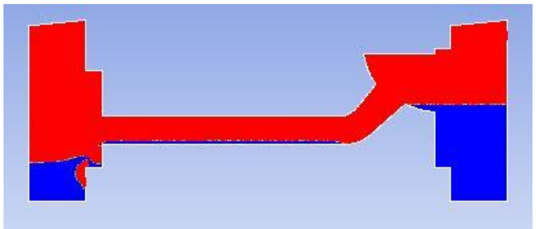
*CASE* ***4.1.1*** *(Changing mass-flow rate of air)*

The phase plots for a set of values of the flow-rates of air have been shown in the subsequent pages.

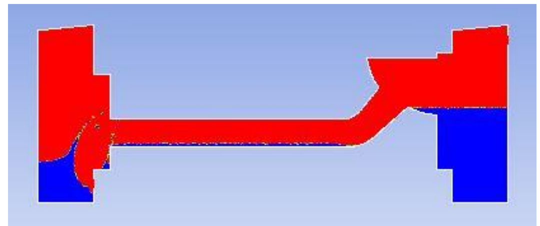
- The time step is taken as $10^{-4}$ seconds.
- The phase plot that has been shown here is taken every 0.1 seconds. This means that the simulation has run for 0.7 s in real time
- Water is represented by blue color whereas air is represented by red color.

The above assumptions are common throughout this chapter. It is very clear from the phase plots given below that changing the flow-rate of mass of air has a dominant effect on the onset of flooding.

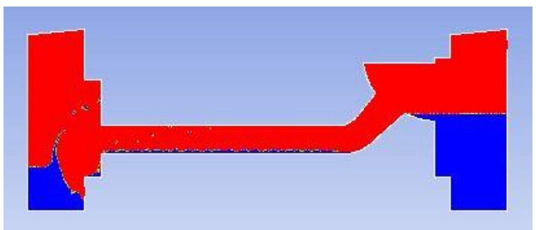
Flow-rate of air = 0.183 kg/s **(Blue – water; Red –air)**

1. 

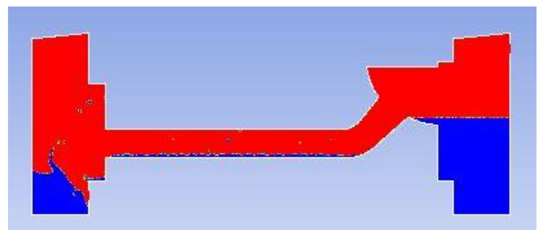
(Time=0.1 second)

2. 

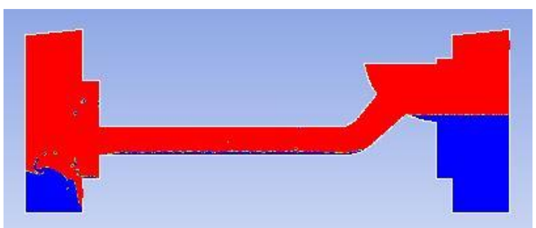
(Time=0.2 second)

3. 

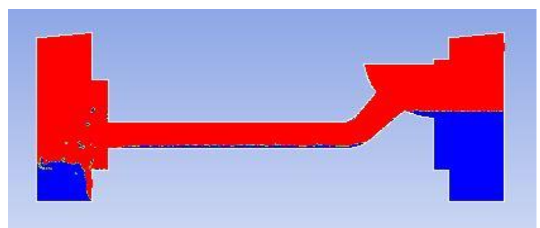
(Time=0.3 second)

4. 

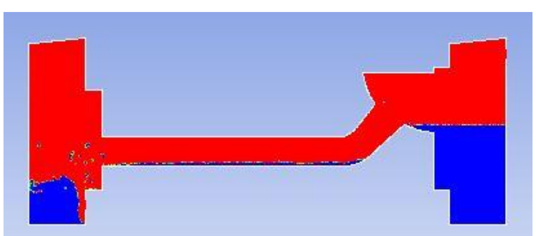
(Time=0.4 second)

5. 

(Time=0.5 second)

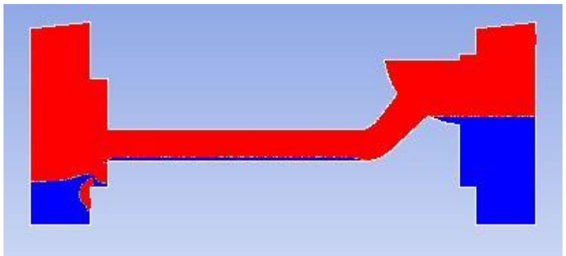
6. 

(Time=0.6 second)

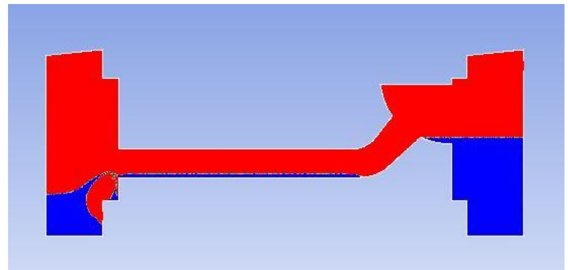
7. 

(Time=0.7 second)

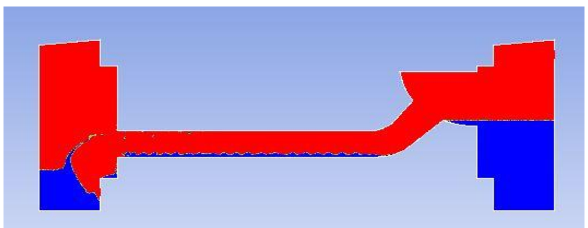
Figure 4.1: Phase plots for flow-rate of air = 0.183 kg/s

1. 

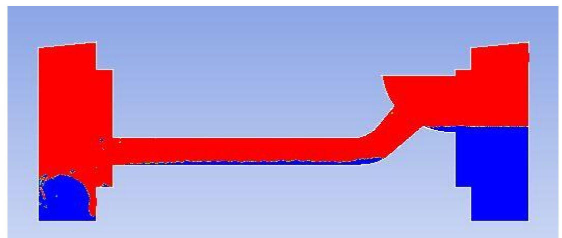
(Time= 0.1 second)

2. 

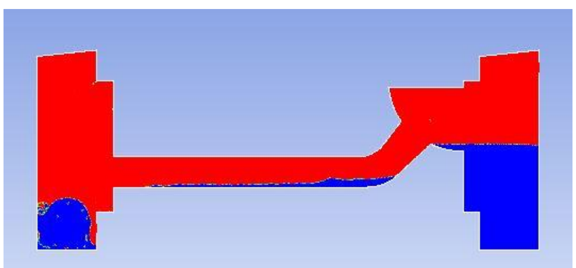
(Time= 0.2 second)

3. 

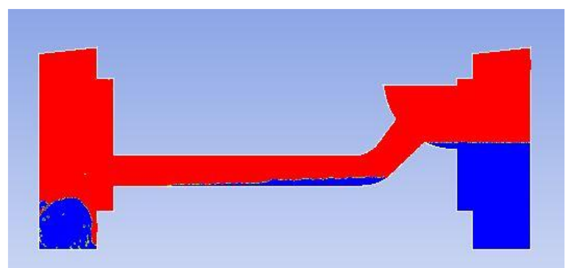
(Time= 0.3 second)

4. 

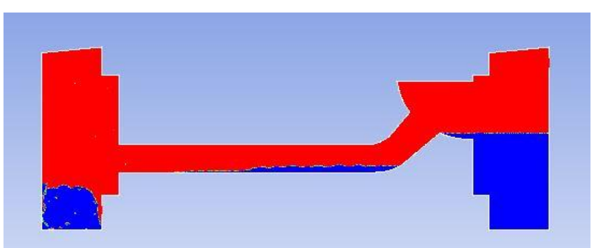
(Time= 0.4 second)

5. 

(Time= 0.5 second)

6. 

(Time=0.6 second)

7. 

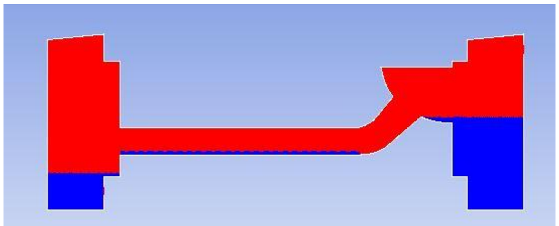
(Time=0.7 second)

Figure 4.2: Phase plots for flow-rate of air = 0.268 kg/s (**Blue – water; Red –air)**

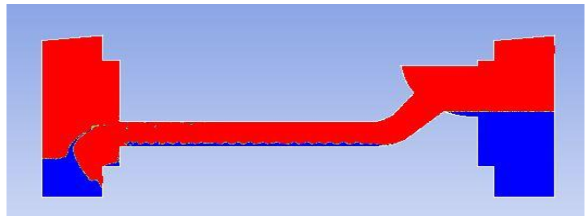
*CASE* ***4.1.2*** *(Changing System Pressure all the other factors being kept constant)*

(System pressure = 0.15 MPa)

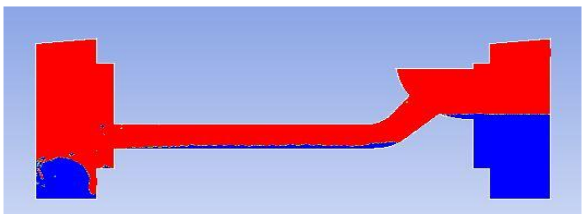
Gauge Pressure = 0.15 MPa (time step = 1 * $10^{-4}$seconds) **(Blue – water; Red –air)**



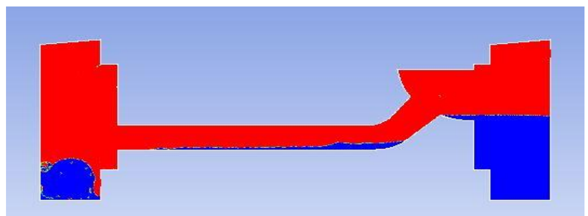
(Time=0.1 second)



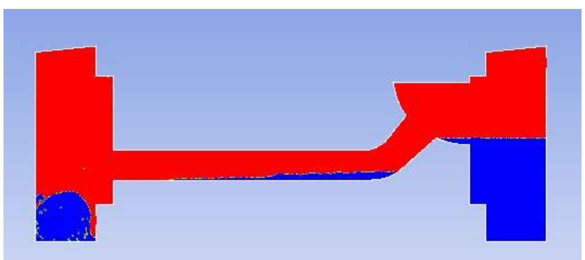
(Time=0.2 second)



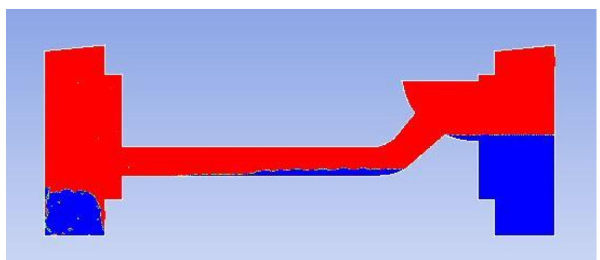
(Time=0.3 second)



(Time=0.4 second)



(Time=0.5 second)



(Time=0.6 second)

Figure 4.3: Phase plots for gauge pressure = 0.15 MPa

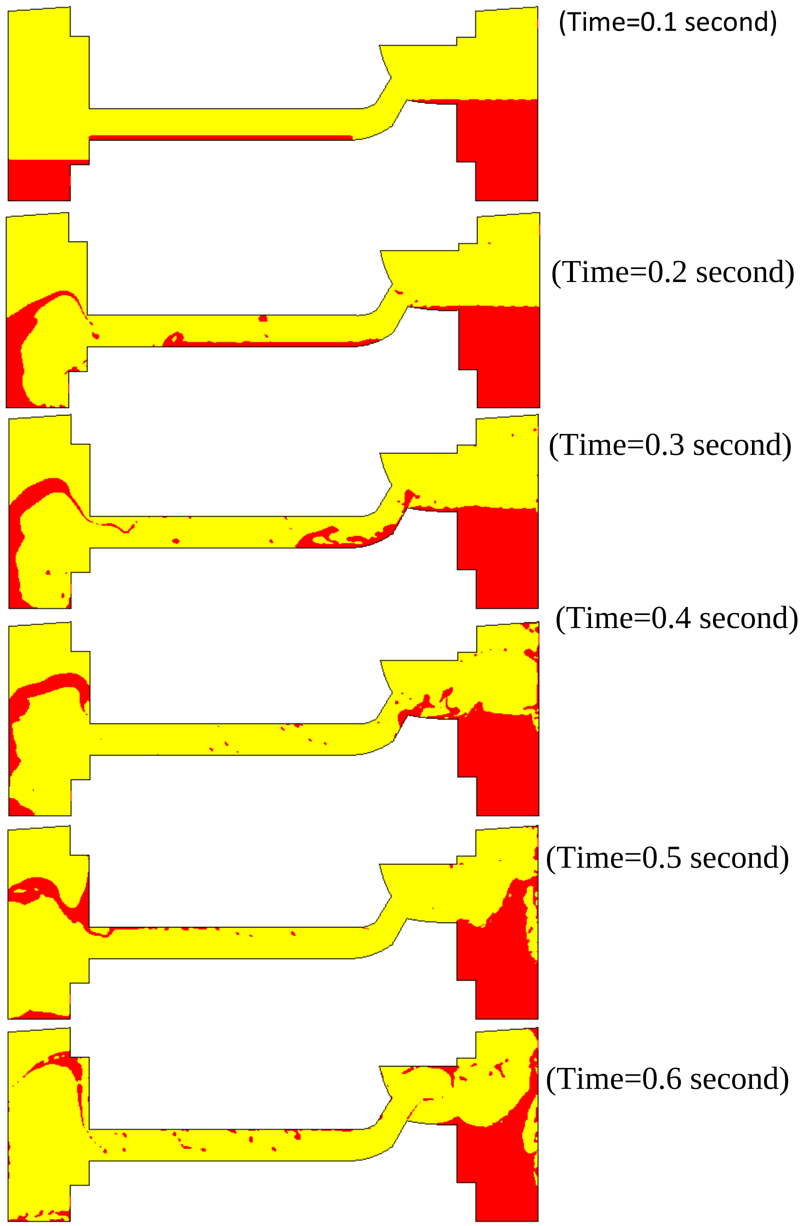



Figure 4.4: Phase plots for gauge pressure = 0 MPa (time step = 1 * $10^{-4}$ seconds; **Yellow- air; Red –water)**

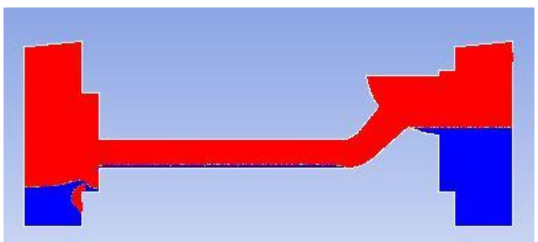
The system pressure has a dominant effect on the start of flooding. Greater the value of pressure in the system more will be the possibility for the onset of flooding in a short time.

## 4.2 **PRESENT WORK**

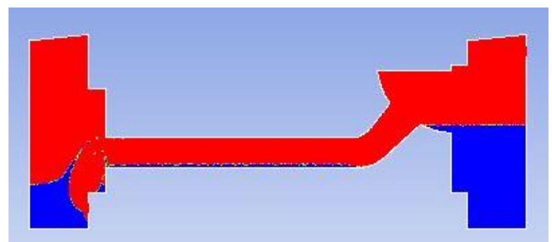
After the validation work has been done, it was evident that no work has yet been done taking the level of water in the hot power leg as a parameter. Also, the effect of varying system pressure has not been studied in detail. Considering the height of the water as a parameter and keeping the flow-rate of air as 0.268kg/s and flow-rate of water as 0.30 kg/s, two cases were obtained in total to study flooding which are discussed below:

### *4.2.1 VARYING THE INITIAL LEVEL OF WATER IN THE HOT POWER LEG*

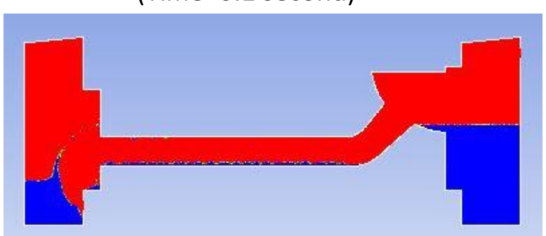
*CASE 1: Initial level of water = 0.53 m* **(Blue – water; Red –air)**



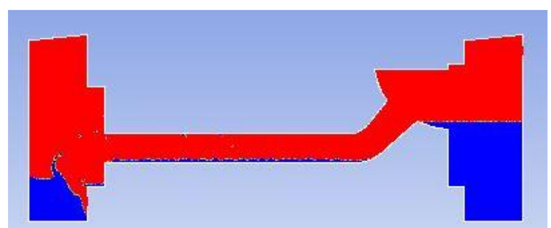
(Time=0.1 second)



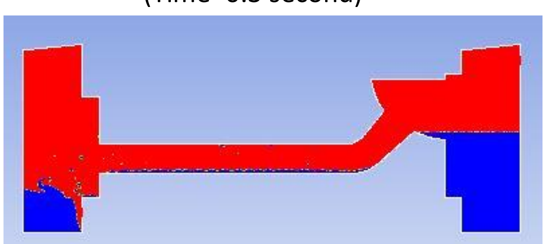
(Time=0.2 second)



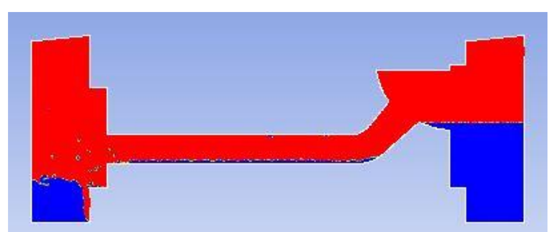
(Time=0.3 second)



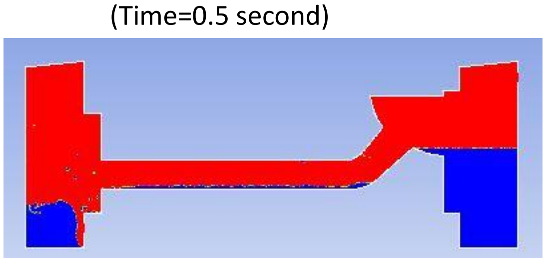
(Time=0.4 second)



(Time=0.5 second)



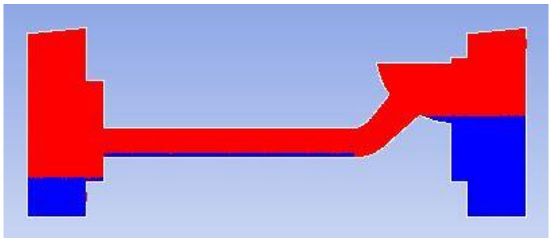
(Time=0.6 second)



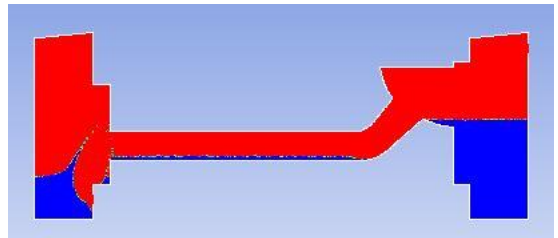
(Time=0.7 second)

Figure 4.5: Phase plots with initial water level = 0.54 m

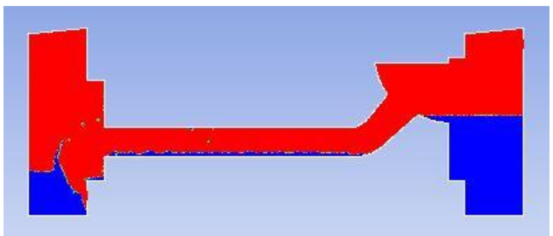
*CASE 2: Initial level of water = 0.54 m* **(Blue – water; Red –air)**



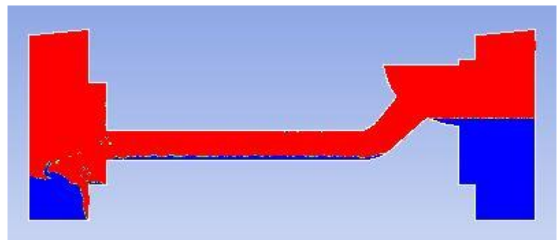
(Time=0.1 second)



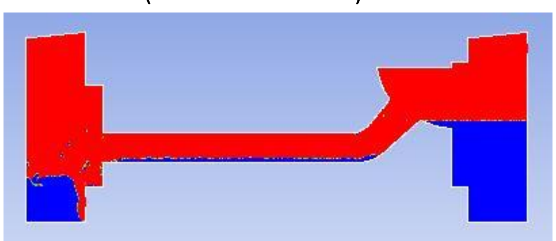
(Time=0.2 second)



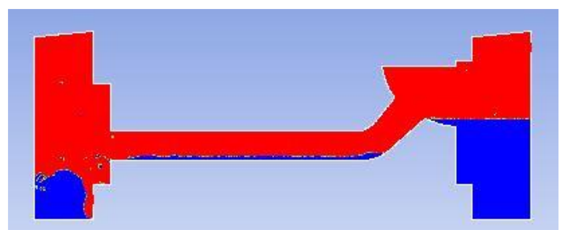
(Time=0.3 second)



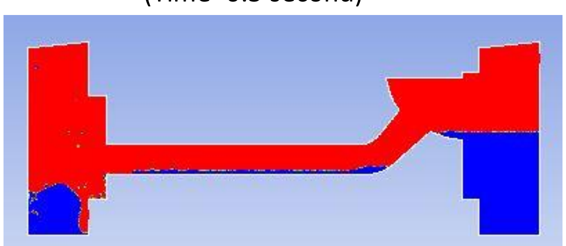
(Time=0.4 second)



(Time=0.5 second)



(Time=0.6 second)



(Time=0.7 second)

Figure 4.6: Phase plots with initial water level = 0.54 m

# CHAPTER-5: CONCLUSION

From the present work, it has been found that

1. As the tallness of the water level increments in the flat part of the hot power leg, there is a more excellent possibility of the onset of flooding.

2. Flooding phenomena relies upon the flow-rate of air. For higher flow-rates of air, flooding was encountered soon enough as contrasted with low flow-rates.

3. The flow rate of air for flooding is directly proportional to the system pressure.

# CHAPTER-6: SCOPE OF THE FUTURE WORK

The height of the water level in the apparatus is just one factor responsible for flooding particularly in case of LOCA for nuclear reactor. There are many other factors like surface tension, rate of flow at inlet of the two fluids, using different fluids other than water and air. The overall attempts made to study flooding would only be fruitful if all these governing factors could be merged into a single parameter valid for most of the cases if not all. So, attempts should be made to generalize these investigations by the means of development of a generalized or empirical parameter for prediction of flooding phenomena which would lead to less accident prone multiphase operations.

# REFERENCES


**Biage, M., Delhaye, J.M.,** 1989, The Flooding Transition: An Appraisal of the Chaotic Aspect of Liquid Film Flow before the Flooding Point. ANS Proceedings, *National Heat Transfer Conf*. 1989:53-60.

**Deendarlianto, Ousaka ,A., Kariyasaki,A., Fukano,T.,** 2005, Investigation of liquid film behavior at the onset of flooding during adiabatic counter-current air-water two-phase flow in an inclined pipe, *Nuclear Engineering and Design*, 235, 2281-2294.

**Deendarlianto, Ousaka,A., Indarto , Kariyasaki,A., Lucas,D., Vierow,K, Vallee,C. and Hogan,K.,** 2010, the effects of surface tension on flooding in counter-current two-phase flow in an inclined tube, *Experimental Thermal and Fluid Science*, 34, 813-826.

**Deendarlianto, Valleea,C., Lucas,D. , Beyer,M., Pietruskea,H. and Carl,H.,** 2008, Experimental study on the air/water counter-current flow limitation in a model of the hot leg of a pressurized water reactor, *Nuclear Engineering and Design* , 238, 3389-3402.

**Drosos, E.I.P., Paras,S..V., Karabelas,A.J.,** 2006, Counter-current gas–liquid flow in a vertical narrow channel-Liquid film characteristics and flooding phenomena, *International Journal of Multiphase Flow*, 32, 51–81.

**Lee, C.S., Bankoff, S.G.,** 1983, A Comparison of Flooding Models for Air-Water and Steam-Water Flow, *Advances in two phase flow and heat transfer*, 64, 745-780.

**Miwa**, **S., Liu,Y., Hibiki, T., Ishii, M., Kondoh,Y., Ukai, N., Tanimoto, N,** 2013, Experimental study of counter-current gas–droplet flow limitation in a 30 cm pipe, *Chemical Engineering Science*, 92, 167–179.

**Nishimoto,T., Tamiya,N., Ami,T., Umekawa,H. and Ozawa,M.,** 2011, Flooding in a Small Diameter Tube (Influence of Tube Material), *Japan Society of Mechanical Engineers*, 4, 1011-1015.

**Ousaka,A., Deendarlianto, Kariyasaki,A., Fukano,T.,** 2006, Prediction of flooding gas velocity in gas–liquid counter-current two-phase flow in inclined pipes, *Nuclear Engineering and Design*, 236, 1328-1292.

**Pantzali,M.N., Mouza,A.A. and Paras,S.V.,** 2008, Counter-current gas--liquid flow and incipient flooding in inclined small diameter Tubes, *Chemical Engineering Science*, 63, 3966-3978.

**Prayitno,S., Santoso,R.A., Deendarlianto, Hohne,T. and Lucas,D.,** 2012, Counter Current Flow Limitation of Gas-Liquid Two-Phase Flow in Nearly Horizontal Pipe, *Science and Technology of Nuclear Installations*, Article ID 513809,9 pages

**Trifonov,Y.Y.,** 2011, Counter-current gas-liquid flow between vertical corrugated plates, *Chemical Engineering Science*, 66, 4851–4866.

**Trifinov,Y.Y.,** 2010 Flooding in two-phase counter-current flows: Numerical investigation of the gas–liquid wavy interface using the Navier–Stokes equations, *International Journal of Multiphase flow*, 36, 549-557.

**Wallis G.B.,** 1969, *One Dimensional Two phase flow*

**Zapke, A. and Kröger, D.G.**, 2000, Counter-Current Gas-Liquid Flow in Inclined and vertical ducts – I: Flow patterns, Pressure drop characteristics and flooding, *International Journal of Multiphase Flow*, 26, 1439-1455.